Numerical analysis of the thermally choked ram accelerator in sub-detonative regime

Tarek Bengherbia[1, 2], Yufeng Yao[2], Pascal Bauer[1], Carl Knowlen[3] and Adam P. Bruckner[3]

[1]Laboratoire de Combustion et de Détonique, UPR 9028 CNRS,
ENSMA, BP 40109, 86961 Futuroscope, France

[2]Faculty of Engineering, Kingston University, Roehampton
Vale, Friars Avenue, London SW15 3DW, UK

[3]University of Washington, AERP, Box 352250, Seattle, WA 98195-2250, USA

1 Introduction

The ram accelerator (RAMAC) is a novel propulsion concept [1]. It uses the shock-induced auto-ignition and flame-stabilized combustion to accelerate the projectile, which travels at supersonic speed in a launch tube filled with premixed combustible mixtures. Detailed investigations have been carried out previously by both experimental and theoretical approaches [1] with three typical propulsion modes (sub-detonative, trans-detonative and super-detonative) having been identified. Of these three detonative modes, one particular interest in recent studies is the thermally choked sub-detonative mode which operates at Mach numbers ranging from 2.5 to 5 and the projectile travels just below the Chapman-Jouguet detonation speed [1] [2]. Figure 1 gives the key flow features associated with this propulsion mode. As the projectile moves at supersonic speed, a system of oblique shock waves is generated. The strong shock-boundary layer interactions cause significant thermal heating on the projectile surface and under certain conditions; propellant ignition can be stabilized by flame holding ability of the bluff projectile base. This will subsequently lead to the combustion of gas mixtures, resulting in significant amount of energy release. The base pressure will increase and the pressure wave propagates upstream generating a stronger normal shock wave on the rear part of the projectile, forming a large subsonic flow region to further sustain the combustion. The flow accelerates to supersonic speed after the thermal choking plane downstream. The combustion at the projectile base generates high pressure and thrust, which accelerates the projectile to very high speeds. While this happens, however, the subsonic combustion itself has a Mach number dependent time scale. In practice, the normal shock wave tends to be either oscillating or monotonically moving upstream/downstream [3]. Furthermore, the choking location will be largely affected by both the large-scale reactive structures presented in the combustion region and the effective shape of the projectile (due to viscous effects). In principle, the inviscid flow models can be used to predict the position of the normal shock-wave and to determine the theoretical bounds of the thermally chocked sub-detonative regime [1]. An earlier study based on one dimensional modeling [4] confirmed that the projectile acceleration and the thermally choked pressure can be determined with fairly good agreement against the test data of the same condition.

2 Numerical Approaches

The presence of oblique shock-waves and their reflections has significant impacts on the auto-ignition and combustion because the latter are very sensitive to the flow conditions [5]. The traditional chemical kinetics

Corresponding author: tarek.bengherbia@lcd.ensma.fr

analysis such as CHEMKIN requires much less simulation time, but it does not consider the detailed gas dynamics of the flow over the projectile. The computational fluid dynamics (CFD) analysis, however, could provide certain in-depth knowledge of the flow and some earlier studies based on CFD simulation of the RAMAC have shown good agreement with the experiments conducted in the subdetonative propulsion mode. Li [6] carried out 3D unsteady reactive flow simulations for a short period of physical time, and the results showed that the sustainability of the subsonic combustion process and the normal shock on the projectile strongly depend on the two key factors: (1) the energy release rate in combustion process, and (2) the separation speed between the obturator and the projectile. Determination of the flow field details requires knowledge of the chemical ignition delay time to get the most accurate results from the CFD analysis, however, the application of a reduced chemical kinetics mechanism can in some cases adequately represent the ignition delay time. Nusca [7] reported that the ignition of the gas mixture can be achieved by the flow stagnation on the projectile back plate (i.e. obturator); this ignition process has also been numerically simulated by Li [8]. Peterson [9] developed two skeletal mechanisms for reactive flow with CH_4/O_2 and H_2/O_2, derived from 190-reactions and 38-species chemical kinetic mechanism. These two models have successfully reproduced the high pressure and the low dilution and have shown significant improvement over the existing reduced methane/oxidation mechanism, particularly for a rich mixture at elevated pressure. The same model was employed by Nusca [10] to successfully simulate the reactive flow over projectiles.

The current research uses a typical RAMAC projectile model, which has been tested at the University of Washington [1], similar to those previous work described in above. The Reynolds-averaged Navier-Stokes equations are solved together with Menter's SST turbulence model [11] for viscous flow prediction. Both steady and unsteady simulations are considered for different configurations (i.e. projectile with and without guiding fins) and at different incoming Mach numbers of 3.5, 4, and 5. Furthermore the numerical solutions of reactive flow are performed for the projectile in sub-detonative propulsion mode, where the projectile has been injected into a stationary tube filled with gas mixture of hydrocarbon fuel (CH_4), Oxidizer (O_2) and diluents (Nitrogen N_2) at fill pressure of 45 atm and temperature of 300K. A variety of reaction models have been adopted, including the eddy dissipation model for finite-rate global and multi-steps chemical kinetics. It is crucial to define the rate constants used in the reduced chemistry model. Instead of using a default value, we obtain this from a separate chemical kinetics code CHEMKIN calculation. After the CFD simulation for obtaining the key parameters such as the combustion volume control and length, the thrust will be calculated by an in-house code TARAM which has been developed at the "laboratoire de combustion et détonique (LCD)" of Poitiers. Numerical results are compared to measured pressure and velocities over the projectile.

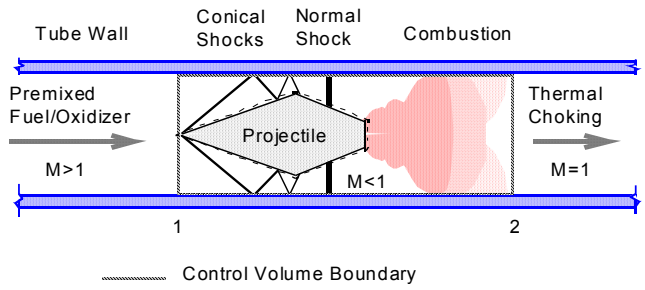

Figure 1. Flow over ram accelerator with thermally

3 Results and discussions

3.1 Computational Procedure of the Flow over Finless Projectile

The computational procedure starts from non-reactive flow over a finless projectile model, i.e. the 'clean' projectile. The study is focused on the problem definition, geometry setting up, multi-block structured mesh generation, and most importantly the grid resolution to identify a baseline mesh for further numerical

simulations. Steady and Unsteady cases are applied for ideal gas equations of state, and no chemical reaction considered at this stage.

3.2 Projectile Configuration and Meshing of Flow Domain

A finless projectile configuration is adopted, which has been widely used in the majority of experimental work to date [1]. The projectile has a bi-conical shape with the conical nose having a cone angle of 25 degree and a length of 52 mm, and the main body cone of a small convergence angle of 1.909 degree and a length of 76mm. The projectile has a maximum diameter of 23 mm, situated at the joint of two cones. The overall length of projectile is thus 128 mm. The computational domain is started from a location of 10 mm ahead of the projectile nose point and ended at a location of 262 mm downstream of the projectile base, resulting in an overall domain length of 400 mm in axial direction. The cylindrical test tube has a diameter of 38 mm. Due to the axi-symmetrical feature of the flow field, only a quarter of the complete cylindrical domain is considered. Same domain will be used even in the case of projectile with four fins. A series of four meshes, of the same topology, has been generated with the aim to identify the baseline grid with the appropriate resolution. Of which, one grid with the minimum grid spacing in the near wall region varies from 0.056 mm to 0.018 mm, resulting in a value between 2 to 24 in wall unit. Hence, it is decided to use this as baseline mesh for the rest of the simulations.

3.3 Simulation of non-reactive flow

Simulations were performed for different incoming Mach numbers (3.5, 4, 5), where free stream pressure and temperature remain constants, 4.5 MPa and 300 K respectively. The aim is to investigate the flow field over the ram accelerator and provide detailed information of pressure and temperature. The peaks in temperature and pressure, as demonstrated in figure 2, move backward with the increase of the Mach number. Figure 3 gives the comparison of tube wall temperature and pressure profiles along the axial direction; the abscissa is the distance from the entrance of the domain.

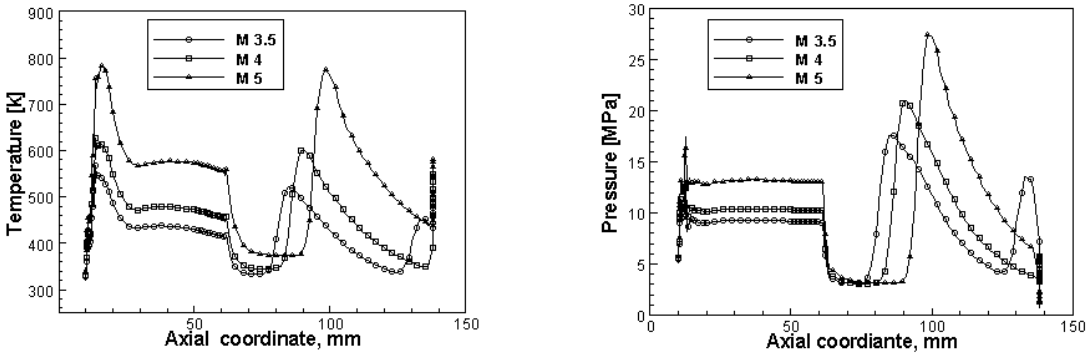

Figure 2: Temperature (a); pressure (b) profiles on the projectile wall.

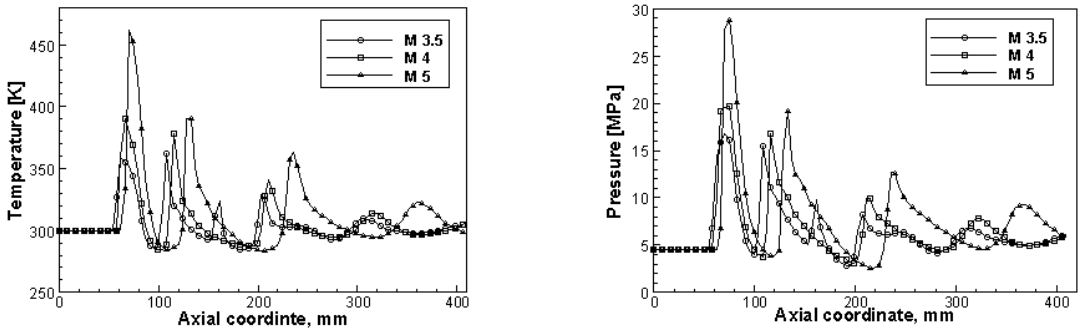

Figure 3: Temperature (a); pressure (b) profiles on the tube wall.

3. 4 Simulation of reactive flow

Simulation continues with the same configuration of projectile, considering the flow field with the fuel/oxidizer gas mixture at a composition of $2.7CH_4 + 2O_2 + 5.6N_2$, that was used in the experimental test, using single step and multi-steps chemical kinetics. Further detailed studies and comparisons with experimental results will be presented in the final paper.

References

[1] Hertzberg. A, Bruckner A.P., Bogdanoff D.W. (1988). Ram Accelerator: A New Chemical Method for Accelerating Projectiles to Ultrahigh Velocities," *AIAA Journal*, Vol. 26, No. 2, pp. 195-203.

[2] Knowlen C (1991). Theoretical and Experimental Investigation of the Thermodynamics of the Thermally Choked Ram Accelerator, PhD Dissertation, University of Washington, Seattle, WA, USA.

[3] Bundy C, Knowlen C, Bruckner A.P (2004). Unsteady Effects on Ram Accelerator Operation at Elevated Fill Pressures, J. of Propulsion and Power, , Vol. 20, No. 2, 2004, pp. 801-810.

[4] Bauer P, Knowlen C, Bruckner A.P (2005). One-dimensional modelling of ram accelerator at high acceleration rates in sub-detonative velocity regime, Eur. Phys. J. Appl. Phys. 29(3), 253-258

[5] Knowlen C, Bruckner A.P, Hertzberg A, Bogdanoff (1991). Operational Characteristics of the Thermally Choked Ram Accelerator, J. of Propulsion and Power Vol. 7, No. 5, 1991, pp. 828-836.

[6] Li, C (1999). Starting process in thermally chocked Ram accelerator, 37th AIAA Aerospace Sciences and Meeting Exhibit. January Reno- Nevada.

[7] Nusca M (1991). Reacting flow simulation for large-scale Ram accelerator, J. of Propulsion and Power Vol. 12, No. 1, pp. 61-69

[8] Li, C (1999). Starting performance in thermally chocked Ram accelerator at high Mixture pressures. 35th AIAA/ASME/SAE/ASEE, Joint Propulsion Conference and Exhibit June 20-23, 1999, Los Angeles, CA

[9] Peterson E. L, Hanson R. K (1996). Reduced mechanisms for Ram accelerator combustion, J. of Propulsion and Power, vol 15, No 4

[10] Nusca M (2002). Numerical simulation of the Ram accelerator using a new chemical kinetics mechanism, J. of Propulsion and Power, vol 18, No 1

[11] Menter F.R (1993). Zonal two Equation k-ω Turbulence Models for Aerodynamic Flows, AIAA 93-2906